Section 9
Stream Water Quality Models

One common objective of water quality modeling studies is to be able to predict the impact of different point and nonpoint source loading scenarios on surface water bodies. Many models predict the impacts of different loading scenarios under different environmental conditions very well. The input requirements for such models, however, are often beyond the scope of many modeling studies. Simpler models may provide acceptable accuracy for a given set of environmental conditions and require less input. The stream water quality models included with BASINS—***QUAL2E*** and ***TOXIROUTE***—provide the user a means of performing simpler modeling studies with somewhat limited data requirements. In situations where a continuous simulation model of the fate and transport of water quality constituents in surface water bodies is required, the BASINS ***Nonpoint Source Model (NPSM)*** can be used.

QUAL2E and ***TOXIROUTE*** are simple one-dimensional, steady-state models. ***QUAL2E*** uses complex algorithms to simulate nutrients, biochemical oxygen demand, dissolved oxygen, temperature, algae, and conservative and nonconservative substances. ***TOXIROUTE*** calculates final and average concentrations of general water quality constituents based on a dilution and first-order decay algorithm. Both models accept point source discharges and are suitable for specific flow conditions. BASINS facilitates the setup of model input by automatically processing geographic and point source data.

The BASINS system also enables the user to view output from these models in a spatial context. The ***Visualization*** tool allows the user to select a classification scheme for visually displaying various flow and pollutant levels in each reach modeled.

9.1 QUAL2E

Purpose

The ***QUAL2E*** model allows users to simulate the fate and transport of water quality constituents in streams under a given flow condition.

Application

QUAL2E is a steady-state,one-dimensional receiving water quality model. Data processing and preparation of an input file for ***QUAL2E*** is automated within the BASINS system. A ***QUAL2E*** simulation includes point source and reach data from BASINS View, as well as any user-supplied nonpoint source data. Some of the BASINS data are tailored, with as few changes as possible, to allow the input file to fulfill ***QUAL2E*** requirements. *QUAL2E Windows Interface User's Manual* (USEPA, 1995) and *The Enhanced Stream Water Quality Models QUAL2E and QUAL2E-UNCAS: Documentation and User Manual* (Brown and Barnwell, 1987) provide further details. Hydraulic structures or dams are not retrieved by the GIS to support the configuration of the stream system selected for simulation.

Procedures

Key Procedures

- ✓ *Activate the Reach File theme*
- ✓ *Select the reaches you want to simulate using QUAL2E*
- ✓ *Select QUAL2E under the Models menu*
- ✓ *Select a year for point source discharge*
- ✓ *Select conservative and nonconservative water quality constituents*
- ✓ *Import QUALINP.RUN using Run under the Import menu in QUAL2E*
- ✓ *View and edit data*
- ✓ *Click on Run to execute QUAL2E*

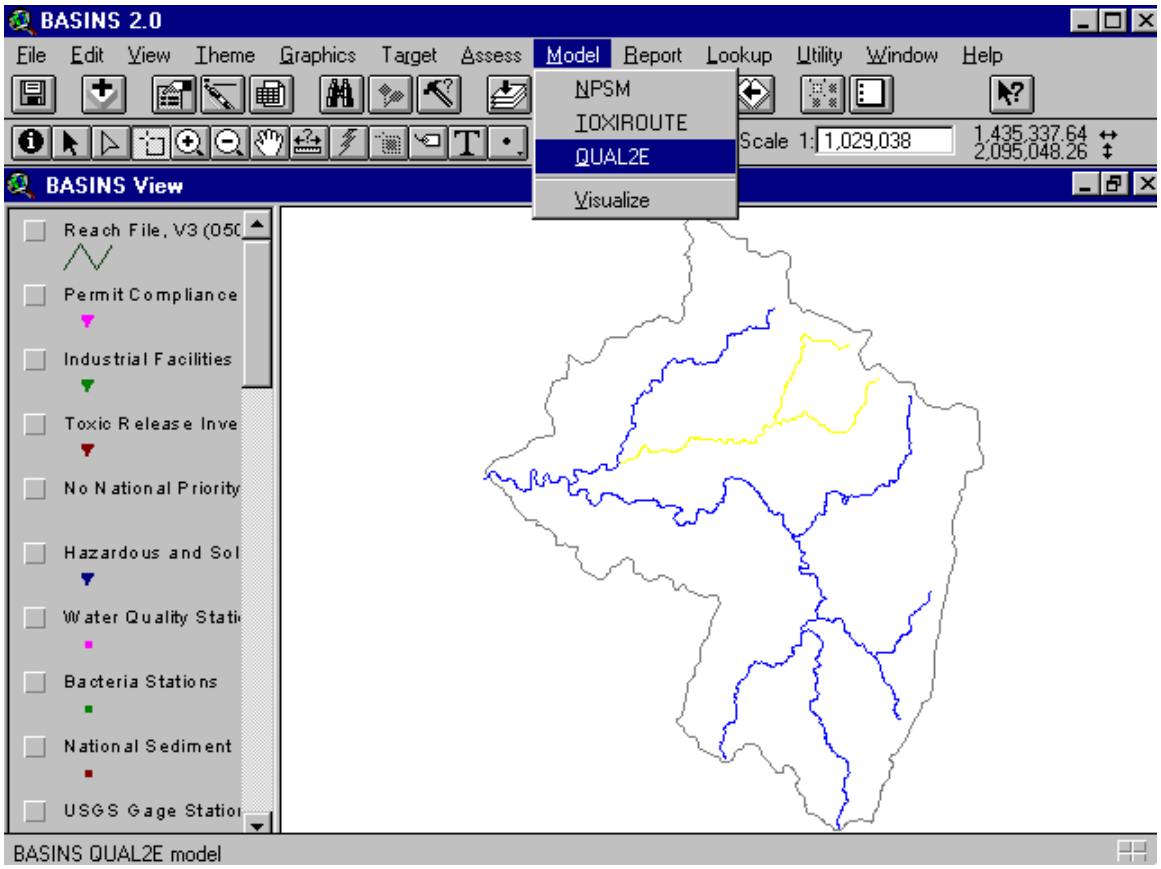
1. In BASINS View, click on the check box next to the Reach File theme and make it active by clicking on the theme legend text. Select the reaches you want to model. In general, remember the following three points when you select reaches:

- Select connected reaches so that they build only one network.
- Do not select too many reaches.
- Reach lengths should not be significantly different.

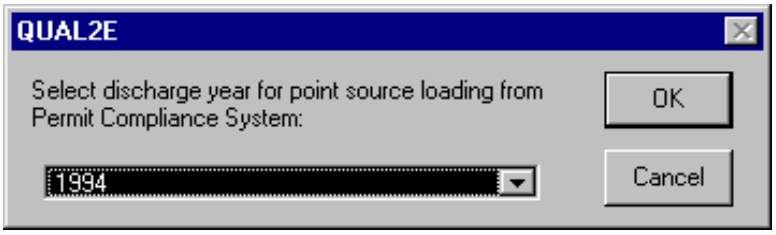
Geographic selection in ***QUAL2E*** simulation is different from that in ***NPSM*** and ***TOXIROUTE*** simulations because individual reaches in a cataloging unit are selected instead of a whole cataloging unit or watershed. BASINS first checks the data pertaining to the selected reaches to find out whether the selected reach network is acceptable for simulation with ***QUAL2E***. BASINS then modifies the reach data slightly such that ***QUAL2E*** reach input requirements are satisfied. ***QUAL2E*** reaches are divided into small computational elements of equal length such that each reach has an integer number of computational elements. In natural systems it is not always possible to find streams that will meet this requirement without adjustment. Therefore, reach lengths in ***QUAL2E*** might appear slightly different from those in BASINS View.

2. Pull down the *Models* menu and select *QUAL2E* (Screen 9.1.1).

3. Select the year of point source data you want to model (Screen 9.1.2).

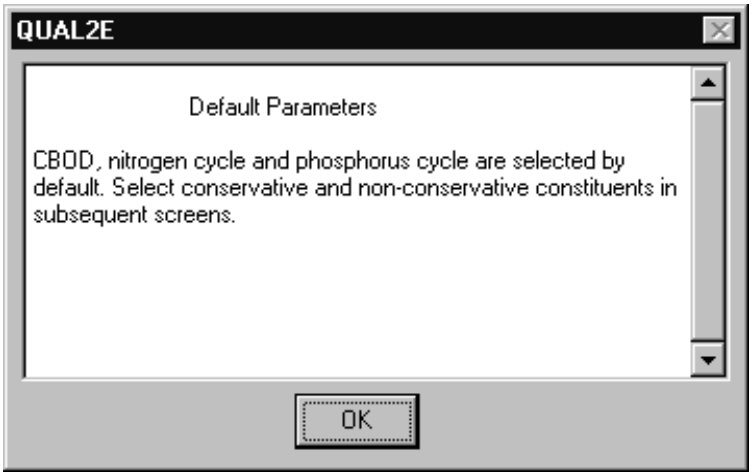


Screen 9.1.1



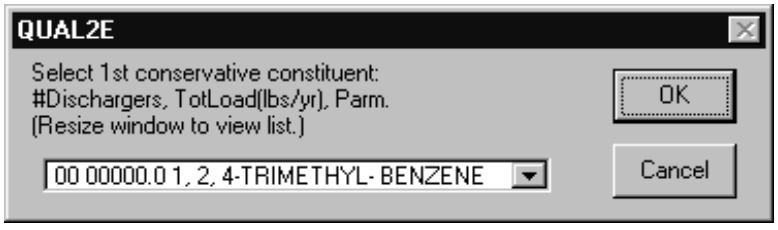
Screen 9.1.2

4. You will be informed that BASINS generates input data for CBOD, dissolved oxygen, fecal coliform, nitrogenous species, and organic and dissolved phosphorus by default (Screen 9.1.3). All of these substances are automatically selected so that any point source data are automatically processed and included in the ***QUAL2E*** simulation.



Screen 9.1.3

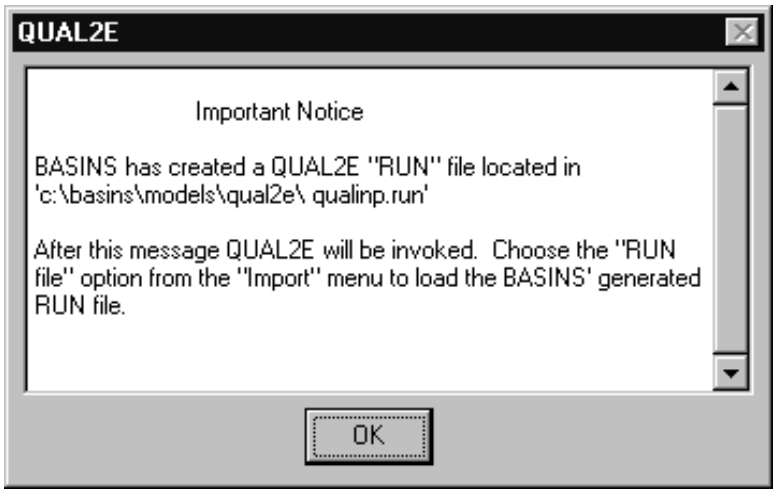
5. You will be prompted to select up to three conservative substances and one nonconservative substance. The numbers in front of the substance name indicate the number of discharges in the selected reaches and the total number of pounds discharged per year. After you select one conservative substance in Screen 9.1.4, click **OK**. You will then be prompted to select a second conservative pollutant. If you do not want to select any more pollutants, click **Cancel**. You will then be asked to select a nonconservative substance.



Screen 9.1.4

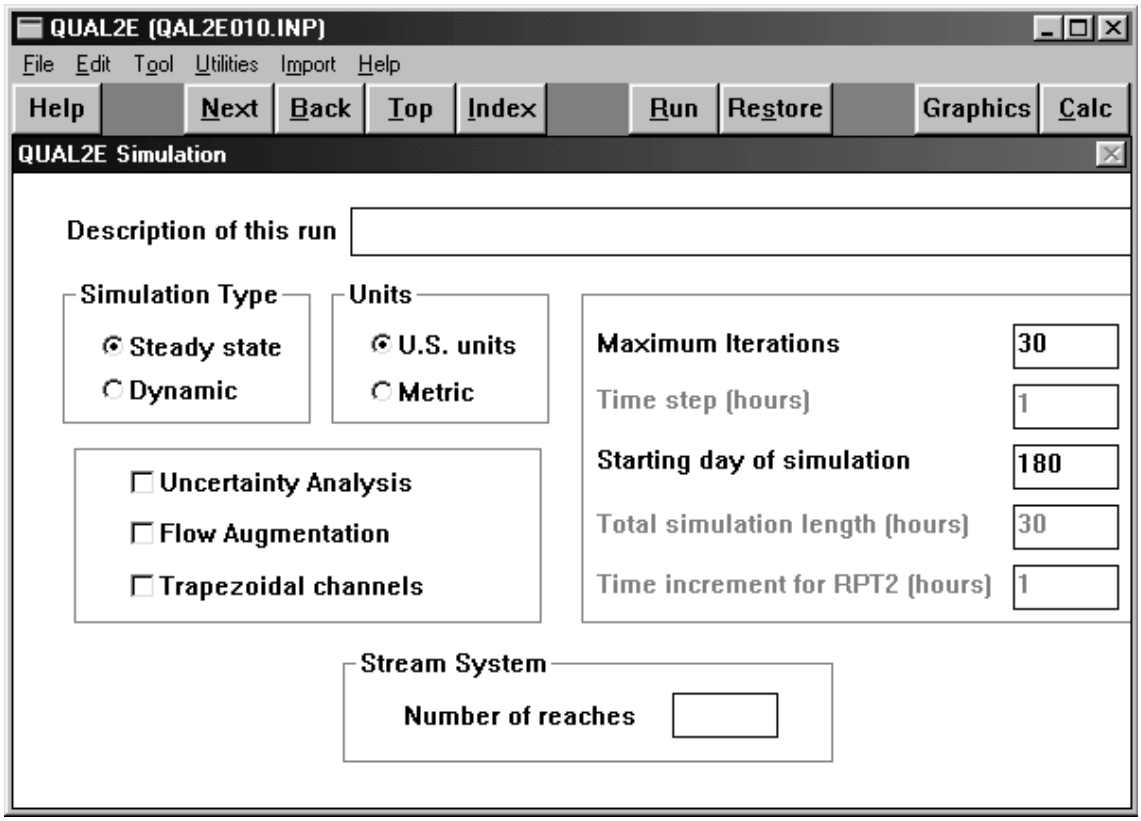
6. Select a nonconservative substance or click **CANCEL** to model only the conservative substance(s) previously selected.

7. You will be notified how to load the input file (QUALINP.RUN) in ***QUAL2E*** (Screen 9.1.5).



Screen 9.1.5

8. You are now in the ***QUAL2E*** graphical user interface (Screen 9.1.6). Choose *RUN file* from the *Import* menu and select QUALINP.RUN to load the input you have just created. You can use the **Next** and **Back** buttons to move from screen to screen. You can click the **Index** button to view a list of all active and inactive screens, and you can go to any screen by clicking the appropriate button. You can modify the data on any of these screens.



Screen 9.1.6

Tip: *If multiple point sources discharge to one computational element, QUAL2E cannot handle them as separate entries. Only one discharger can be specified per computational element, forcing BASINS to total all the dischargers in a computational element while preparing a QUAL2E input file. Therefore, you might see "5 disch" (five dischargers together) or "3 disch" instead of a discharger name in the Point Loads and Withdrawls screen in QUAL2E.*

Tip: *When some of the necessary information is not available in BASINS View, a reasonable value is assigned to fill the blank (for example, the temperature of point source discharges is assumed to be 25°C). Some additional information is stored in DEFAULT.Q2E, which can be viewed and modified using any text editor. These data are also used to prepare an input file.*

Tip: *In a **QUAL2E** simulation, if the user selects a reach that has an upstream reach not included in the current simulation, BASINS assumes that the selected reach is a headwater reach while preparing the input file for **QUAL2E**. If the user wants to carry over the effect of upstream discharges, he or she might have to model upstream reaches separately, record the output flow and concentrations, and type these numbers in the Headwater Source Data screen in **QUAL2E**.*

Tip: *By default QUAL2E uses 7Q10 flow as stream flow. You may change stream flow values to simulate other conditions (e.g., mean flow).*

Tip: *QUAL2E uses its own numbering scheme for reaches. The reach number in the Reach File 1 (RF1) database appears in the Reach Name column of Screen 2 (Stream Reach System).*

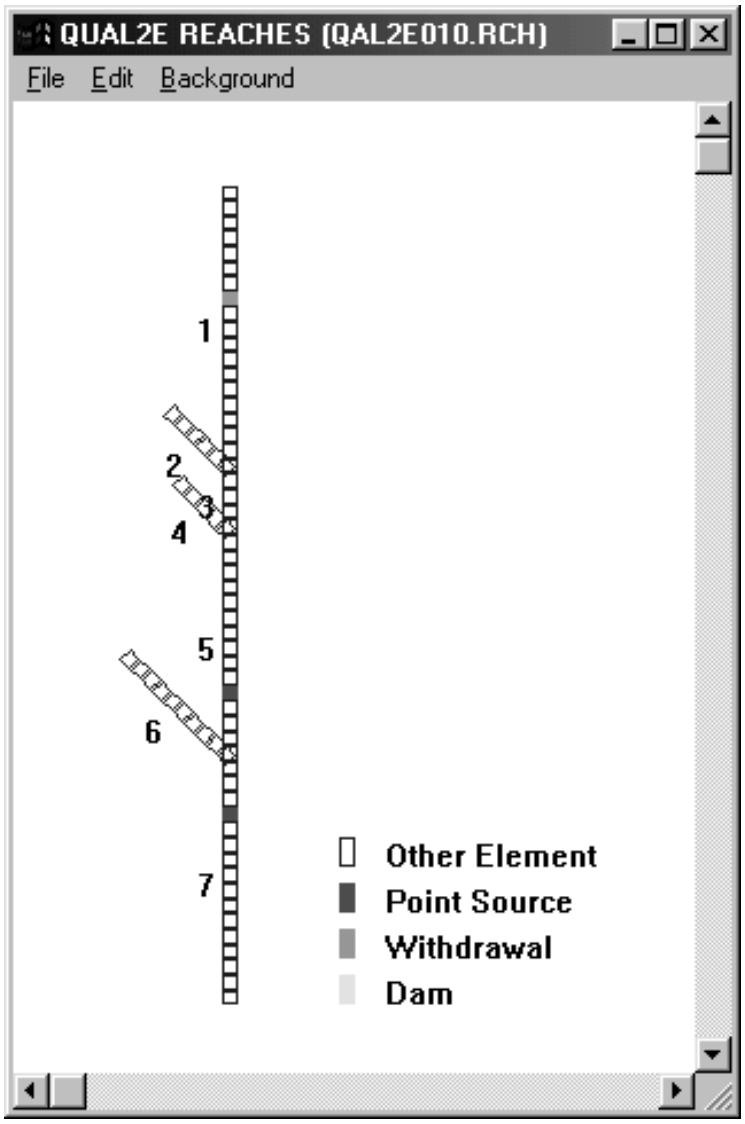
Tip: *When a discharger is located at the most upstream or most downstream computational element of the reach, the discharger location is shifted one computational element inside the reach because **QUAL2E** does not allow these two elements to receive point source discharges.*

Tip: *The rating curve information for stream reaches is not generated automatically. You will have to develop rating curves for all the reaches you are simulating and substitute the values in Screen 11 (Hydraulic Data).*

9. Save the project. Click **Run** to execute the model.

10. You can view the output file after the model has run or you can view the output in BASINS View by selecting *Visualize* from the *Models* menu. Refer to Section 9.3 for more information on the ***Visualization*** feature. Click **Yes** when you are asked if you want to view the model output. BASINS will display the model output using a text editor. Close the window when you are done. For more information on using QUAL2E and the Windows interface to ***QUAL2E***, see the references (Brown and Barnwell, 1987; USEPA, 1995). In addition, the ***QUAL2E*** Windows interface user's manual is included, in PDF format, in your BASINS installation under \BASINS\MODELS\QUAL2E\USERMANL.

11. Click the **Graphics** button to begin the plotting program. Click the **REACHES** button. This screen (Screen 9.1.7) displays the reach network and the computational elements used for the simulation. Note the locations of point sources. Click **File** and **Exit** to exit this screen.



Screen 9.1.7

TUTORIAL

- *In BASINS VIEW (TUTORIAL.APR file), display and activate the REACH FILE theme. Locate and select the following reaches:*

  05010007012

  05010007013

  05010007014

- *Select 1993 as the point source discharge year.*
- *Click CANCEL when prompted for a conservative substance.*
- *Once the QUAL2E Windows interface appears, select the Import menu and choose Run. Recall the name of your input file is QUALINP.RUN. Select this file. Performing this action loads data from BASINS into QUAL2E.*
- *Since there is no point source discharging any pollutant (according to PCS) in the three stream reaches being modeled, all the elements in Screen 3 (Computational Elements) are standard elements and Screen 17 (Point Loads and Withdrawals) is inactive. Assume that you know there is a point source facility discharging at 9.2 kilometers upstream of the confluence of reaches 05010007013 and 05010007014. Also assume that you have the following information.*

  Point source facility name : Blacklick WWTP

  Receiving stream reach id (Cuseg) : 05010007013

Treat (%)	*0.0*
Flow (m3/s)	*0.5*
Temperature (°C)	*23.0*
DO (mg/L)	*5.0*
BOD (mg/L)	*50.0*

  In QUAL2E each reach is divided into multiple cells called computational elements. The length of each computational element is 1.642 km (Screen 2, Stream Reach System). Therefore, Blacklick WWTP is located at the 7th computational element of reach 05010007013. Note that reach 05010007013 becomes Reach No. 1 in QUAL2E. Change the type from S to P for the 7th element of Reach No. 1 in Screen 3 (Computational Element). Uncheck all the boxes except BOD and DO in Screen 4 (Water Quality Simulation). Open Screen 17 (Point Loads and Withdrawals), and you will find a blank line in the table. Enter the facility name as Blacklick WWTP. Also enter flow and concentrations information as shown above.

- *Click on the **Index** button and open Screen 11 (Hydraulic Data). Data elements, except for Manning's roughness coefficient, in the Hydraulic Data screen are not available from the RF1 database. Therefore, you will have to supplement the data from other sources. Assume depth versus flow and velocity versus flow relationships are known. Using Figures 9.1.1 and 9.1.2, substitute the coefficients and the exponents in appropriate fields of the Hydraulic Data screen.*
- *Convert them to cubic meters per second by dividing the numbers by 35.3. Now compare these numbers with flows listed in the Headwater Source Data screen of QUAL2E. The default setup is to simulate low flow condition, but you may change the values to simulate mean flow condition.*

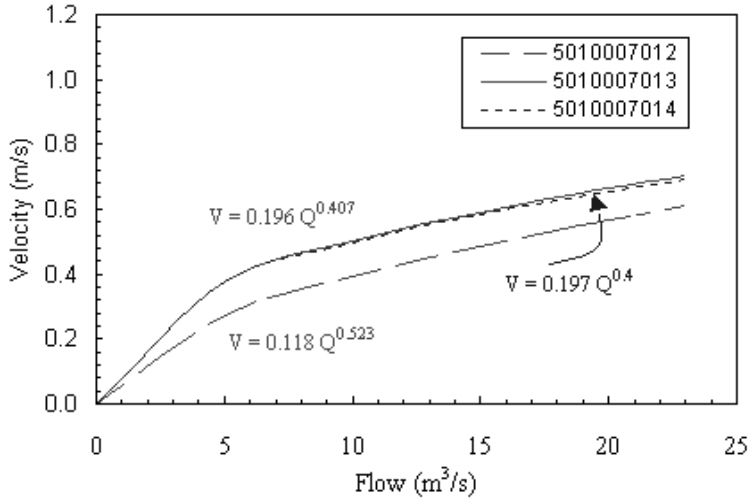


Figure 9.1.1

- *Open Screen 16 (Headwater Source Data) and write Headwater Names and corresponding flow values*

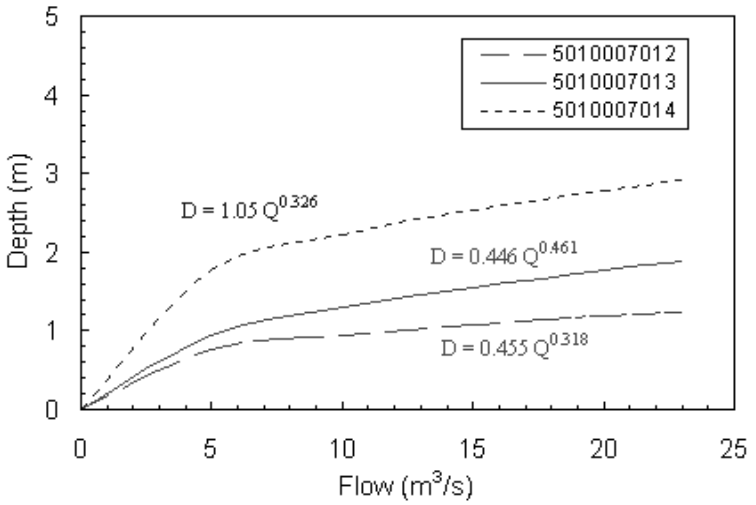


Figure 9.1.2

(note the units). Open BASINS ArcView and activate Reachfile 1 theme. Select the i tool and click on the reaches you are simulating. Write MNFLOW (mean flow) and SVTNFLOW (low flow) values of reaches 05010007013 and 05010007014. These flow values are in cubic feet per second.

- *To plot a graph, select 1 as Starting Reach and 3 as Ending Reach. Note that the Flow versus Distance option is selected as the Type of graph. Click* ***Run*** *and press* ***OK*** *to accept that the plot will be saved as the selected file. The graph shows a plot of flow versus distance. You may plot water quality constituents versus distance. Select 1 as Starting Reach and 3 as Ending Reach in the first graphics screen. Select the Water Quality Constituents versus Distance option selected as the Type of graph and click* ***Next****. Select Biochemical Oxygen Demand and Dissolved Oxygen. Click* ***Run*** *and press* ***OK*** *when you are asked to confirm the filename in the Graphics Selection screen. Close the plot window when you are done. Also close the QUAL2E Graphics window.*

9.2 TOXIROUTE

Purpose

TOXIROUTE provides a modeling tool for BASINS users to perform simple assessments of pollutant concentrations in rivers.

Application

TOXIROUTE uses a simple first-order decay solution to simulate the transport of selected pollutants in streams and rivers. This simplification provides an initial approach for examining concentrations of discharged pollutants in receiving waters. ***TOXIROUTE*** does not explicitly consider nutrient or chemical reactions or transformations. In cases where algal growth or other significant chemical processes are a concern, this simplified model might be inappropriate. The ***TOXIROUTE*** model assumes steady-state conditions, where the system has reached equilibrium. The methodology might have limitations in cases where wet weather processes, such as nonpoint source runoff, predominate. When the ***TOXIROUTE*** model is applied within BASINS, the model receives point source discharge and reach data from BASINS View.

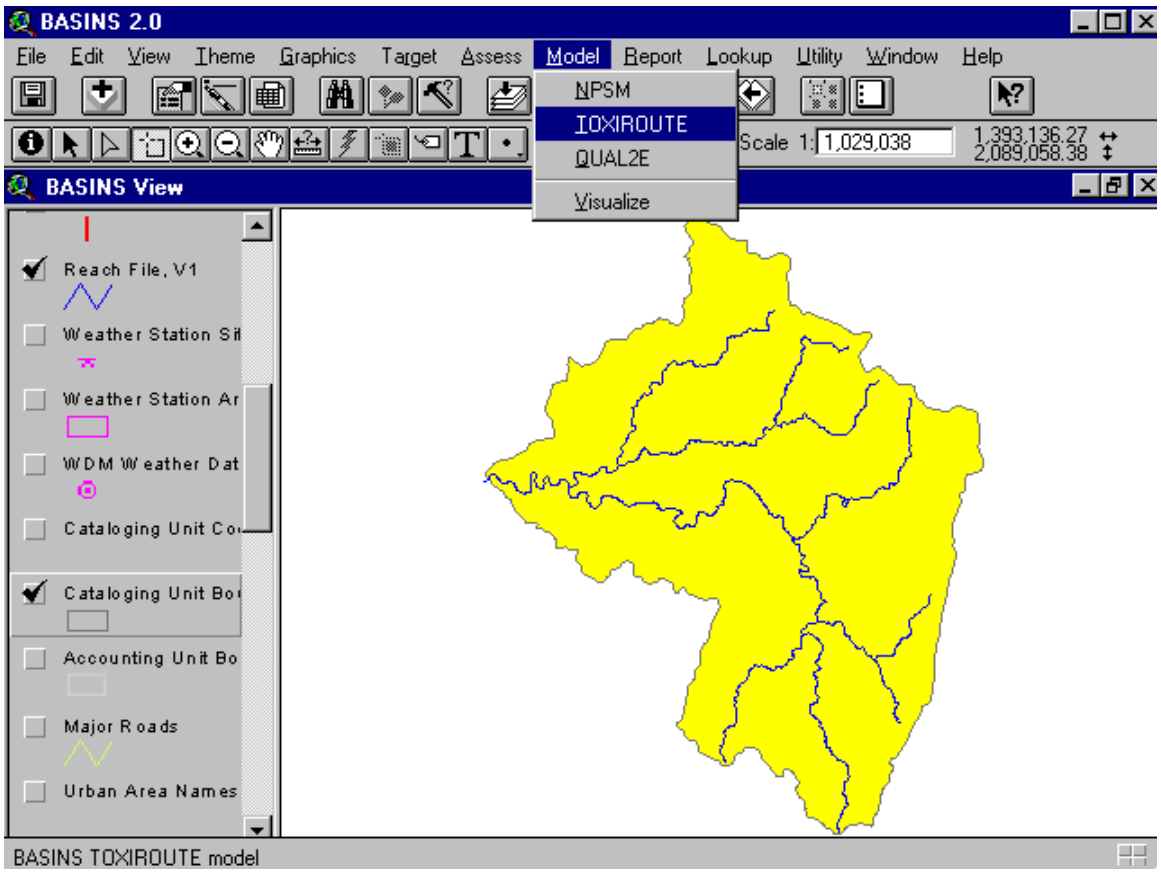
Procedures

Key Procedures

- ✓ *Activate the Cataloging Unit Boundary theme in BASINS view*
- ✓ *Select a cataloging unit*
- ✓ *Select Toxiroute under Models menu*
- ✓ *Select a year for point source discharge data*
- ✓ *Select a pollutant*
- ✓ *View and edit data, if necessary*
- ✓ *Click Run to execute the model*

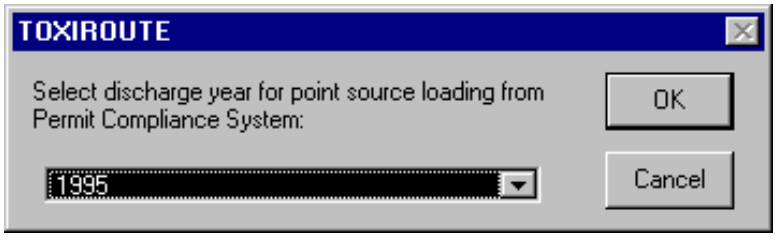
1. In BASINS View make Cataloging Unit Boundary the active theme and select a cataloging unit.

2. Pull down the *Models* menu and select *TOXIROUTE* (Screen 9.2.1).



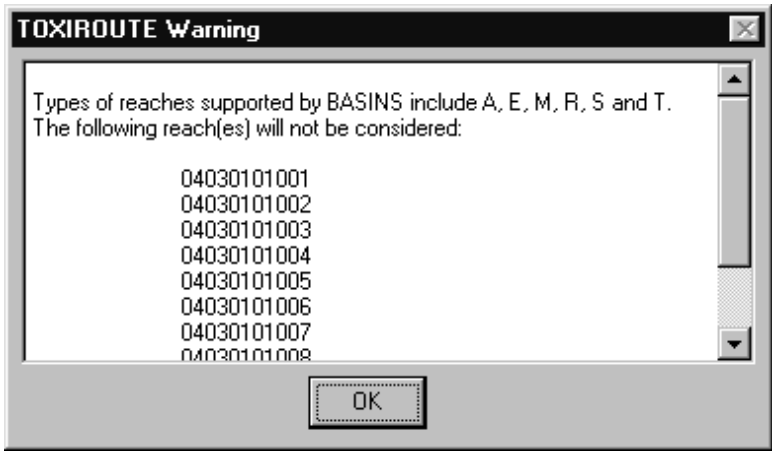
Screen 9.2.1

3. Select the year of point source data you want to model (Screen 9.2.2).



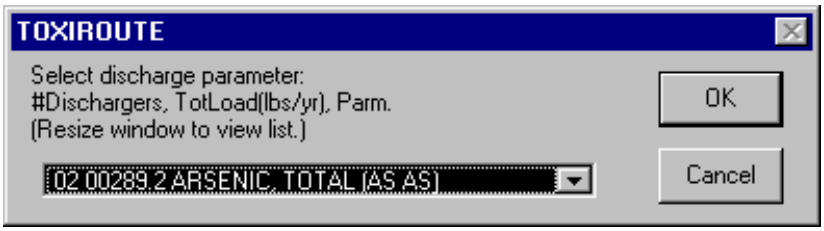
Screen 9.2.2

4. ***TOXIROUTE*** will only simulate pollutant transport in certain reach types including A, E, M, R, S, and T. The reach type is included in the Reach File, Version 1 database. If the selected study area contains other reach types, a ***TOXIROUTE*** warning dialog box will appear (Screen 9.2.3). This dialog box includes a list of the reach segments that are not be considered in the simulation. Click **OK** to continue.



Screen 9.2.3

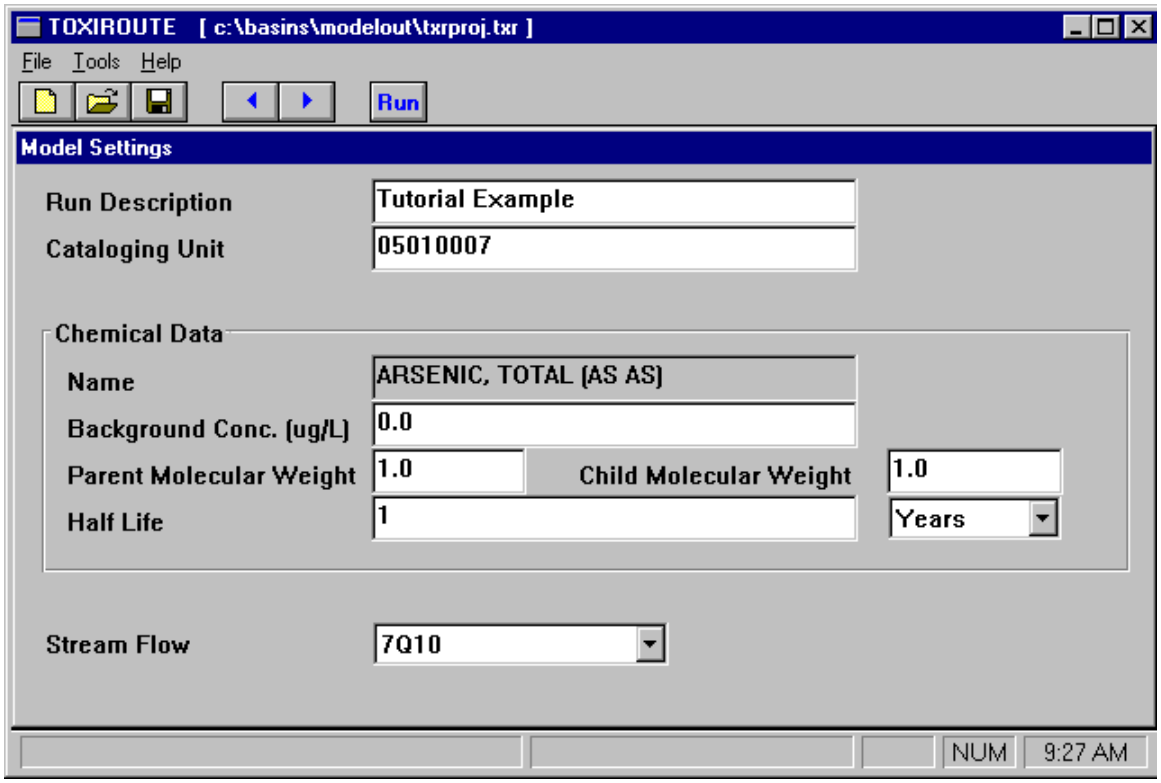
5. You will be prompted to select a pollutant from a list of pollutants (Screen 9.2.4). If available, BASINS View generates the point source data for the selected cataloging unit. ***TOXIROUTE*** automatically loads the information provided by BASINS View. To help you select a pollutant, the two numbers before the pollutant name indicate the total number of facilities in the cataloging unit and the total estimated load of all discharges combined in pounds per year, respectively.



Screen 9.2.4

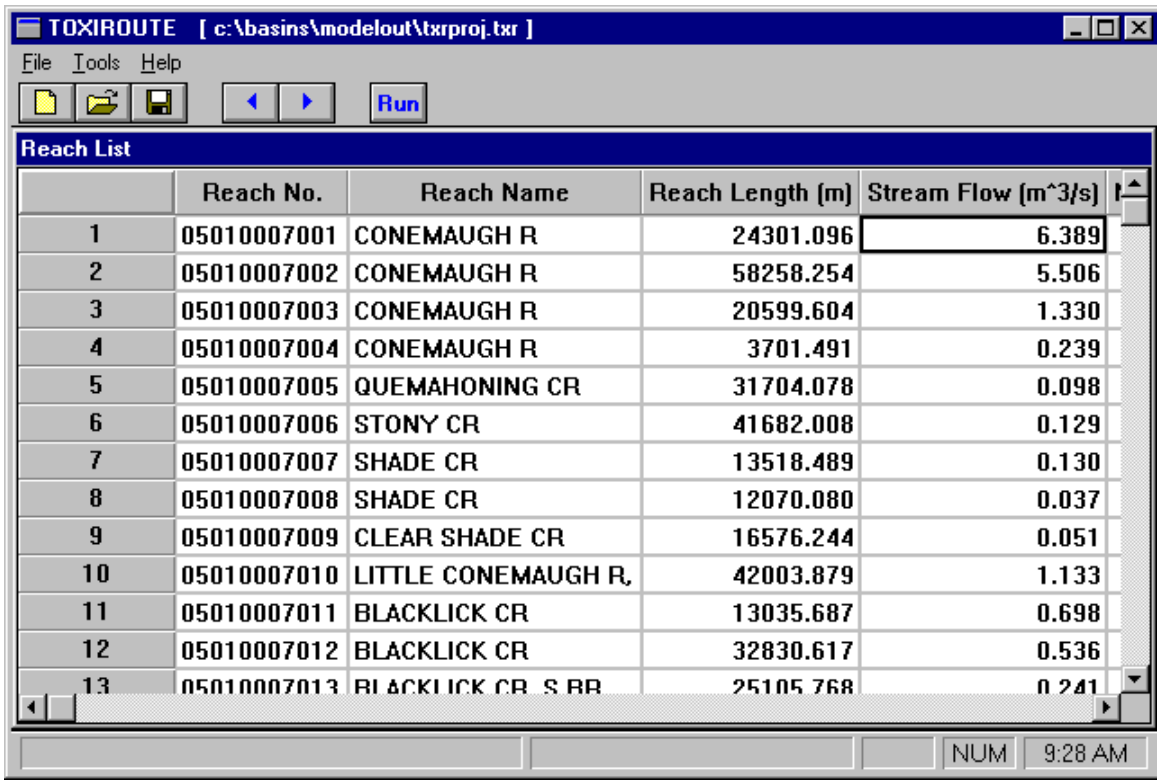
6. The first screen of ***TOXIROUTE*** (Screen 9.2.5) allows you to select pollutant-specific parameters for simulation. There are six buttons immediately below the menu bar of each screen. If you place the cursor on any of these buttons, you will see a brief description of the button in the status bar at the bottom left corner of the screen. The buttons are, from left to right, **open a new project file (TXR file)**, **open an existing project file**, **save the project file**, **go to the previous screen**, **go to the next screen**, and **run the model**.

7. For the selected pollutant, enter the background concentration, parent molecular weight, child molecular weight, and half-life. Background concentration refers to the concentration observed at the upstream end of simulated reaches. Available monitoring data are typically used to estimate the background concentration. The parent molecular weight and child molecular weight are not significant if there is no degradation product (child chemical) produced during the decay of the selected pollutant (parent chemical) (USEPA, 1985). The parent and child molecular weight are used to calculate the child chemical concentration. The stream flow selection box lets you select 7-day 10-year low flow (7Q10) or mean flow. The 7Q10 is representative of dry summer conditions based on a statistical analysis of flow data. Click the right arrow to go to the next screen.



Screen 9.2.5

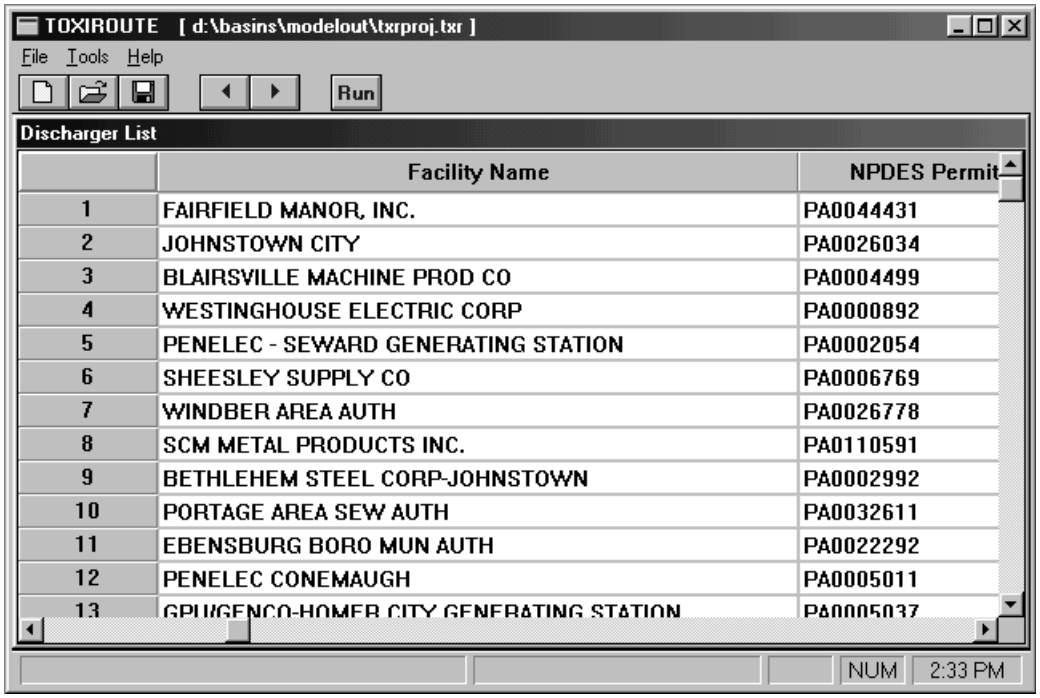
8. In the Reach List screen (Screen 9.2.6), you can view information about all of the reaches in the cataloging unit, including their lengths and stream flows. Currently the NPS flows and NPS loads



Screen 9.2.6

columns are not active. Use the horizontal and vertical scroll bars to view other parts of the screen. You cannot edit any data on this screen. Click the right arrow to move to the next screen.

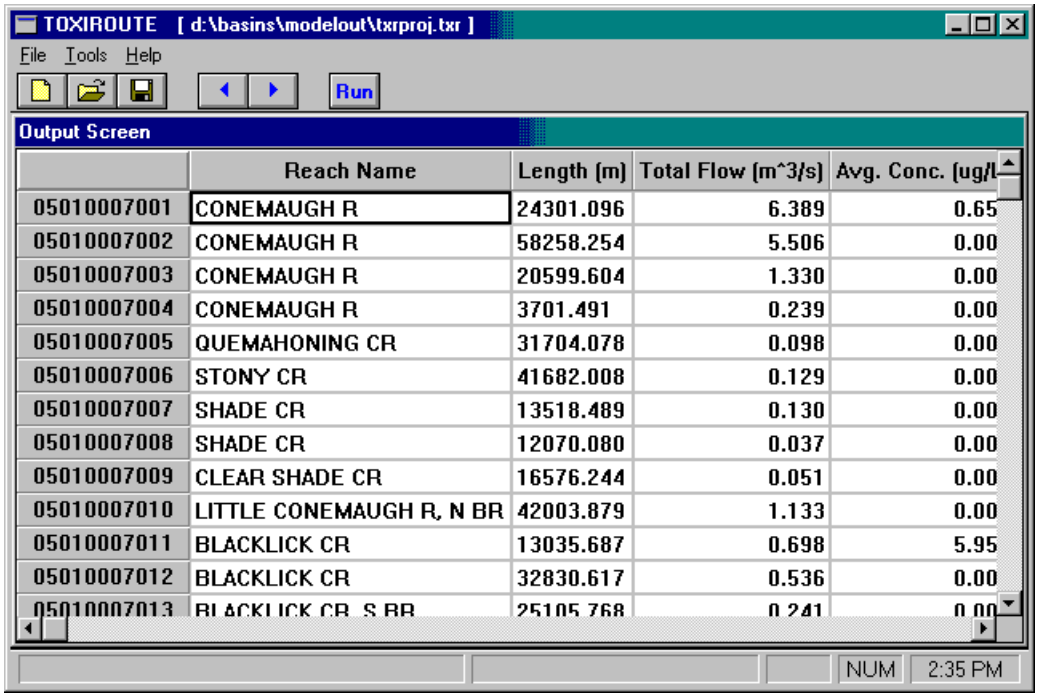
9. In the Discharger List screen (Screen 9.2.7), you can view/edit point source loading information. All of the facilities (or dischargers) in the cataloging unit are listed regardless of their discharge of the particular pollutant. You can edit the load limits from a facility by clicking on the cell and typing a number. If a discharger is located at the most downstream point of a reach (the distance of the discharger location from the endpoint of the reach is 0.0), ***TOXIROUTE*** assigns the lesser of 1 meter or 1 percent of the reach length as the distance from the bottom of the reach.



Screen 9.2.7

You can also add or delete a facility by clicking the right mouse button once. When you click the right mouse button, a pop-up menu appears with two options—*Add Discharger* and *Remove Discharger*. If you choose to add a discharger, a blank line will appear at the bottom of the table. By default the model will assign a value of 0.0 to the distance and load columns and will specify the type as Discharger. You can edit the load and distance. After you enter data in any column for the new discharger, press Enter/Return to accept. BASINS will then automatically activate the next cell to the right. If you want to delete a facility, make any cell in the row of the facility active (click the left mouse button on the cell) before you click the right mouse button and then select *Remove Discharger* from the pop-up box. This is the last screen for ***TOXIROUTE*** input. You can run the model by clicking on **Run**.

10. The Output screen (Screen 9.2.8) lists concentrations on a reach basis. The Average Concentration column lists average concentrations of the pollutant in reaches, whereas the Final Concentration column lists the concentrations of the pollutant at the end of the reaches. The average concentration of a reach is the mean value integrated for the total length of the reach. The final concentration is the concentration at the downstream end of the reach. It should be noted that when one discharge or several discharges are located at the extreme end of a stream reach, the resulting final concentration of this reach is usually significantly higher than the average concentration due in large part to the



Screen 9.2.8

location of the discharges. The Child Concentration column shows the final concentrations of the chemical produced during the decay of the parent chemical. Use the scroll bars to view the hidden parts of the screen. You need not save the output file. Each time you run the model, it generates an output file with the project name as the file name and .out as the filename extension. You can save the output under any name by selecting *Save Output* under the *File* menu. You can view the output by selecting *Visualize* under the *Models* menu in BASINS View. Refer to Section 9.3 for further details.

9.3 Visualization

Purpose

With ***Visualization***, the user can view model output graphically in BASINS' GIS environment. Streams are displayed in different colors to portray the relative magnitude of flow and pollutant concentrations.

Application

Visualization is a tool included in BASINS to allow the user to view the model results in the BASINS View environment. In this way the model results can be interpreted spatially. Using this approach, the user can concurrently view other relevant data, such as land use, point source locations, and gaging stations. Currently, BASINS supports visualization of the ***TOXIROUTE*** and ***QUAL2E*** simulation results. ***NPSM*** output cannot be viewed in BASINS.

Procedures

Key Procedures

Select Visualize under the Models menu.

Select a model name

Select an output file

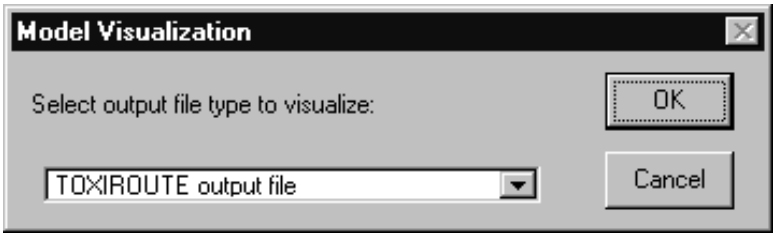
Select or edit the classification scheme when prompted

Visualizing TOXIROUTE Output

TOXIROUTE simulates one pollutant for all reaches in a cataloging unit during each application. The model output contains four elements—average concentration, final concentration, child concentration, and stream flow. In BASINS View, you can choose one of the four elements to visualize.

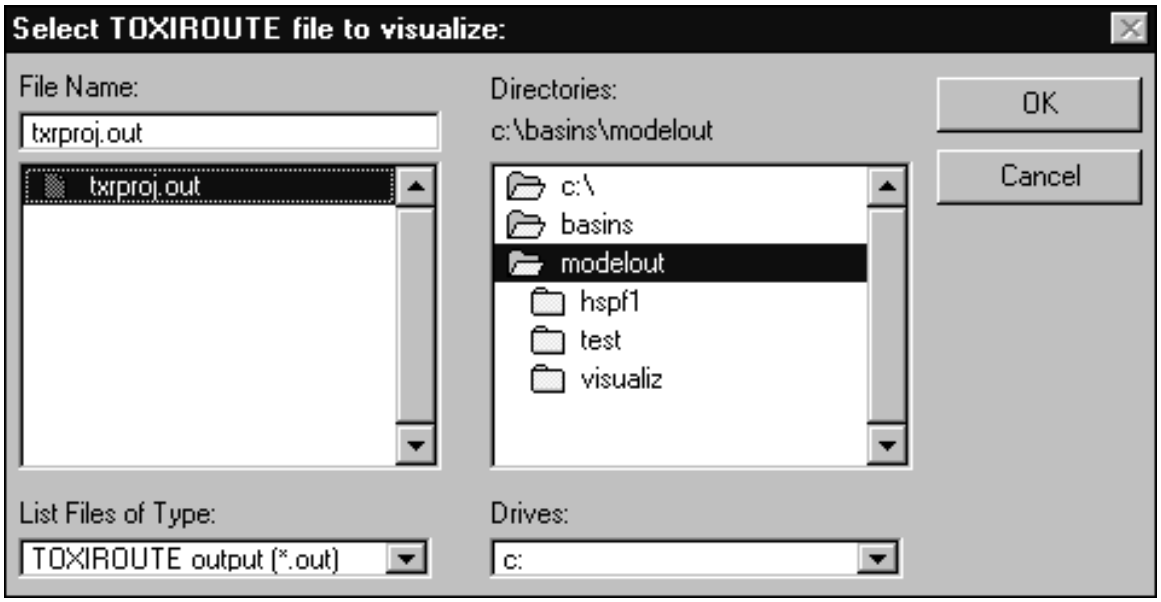
1. In BASINS View pull down the *Models* menu and select *Visualize*.

2. Choose "TOXIROUTE output file" in the output file type selection dialog box (Screen 9.3.1).



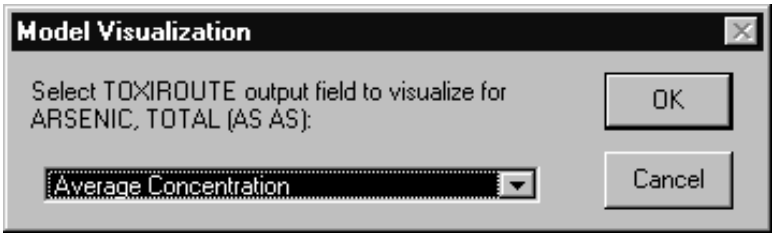
Screen 9.3.1

3. Specify the ***TOXIROUTE*** output file name in the file selection dialog box (Screen 9.3.2). The ***TOXIROUTE*** output files are located in the \BASINS\MODELOUT directory. An output file has the same name as the ***TOXIROUTE*** project name, but it has an .out extension. When a file is created in the ***TOXIROUTE*** output screen, you have the option to choose any name for an output file by selecting the *Save Output* option under the *File* menu. The default project name is TXRPROJ, and therefore the default output file name is TXRPROJ.OUT. After you have selected the output file name, click **OK**.



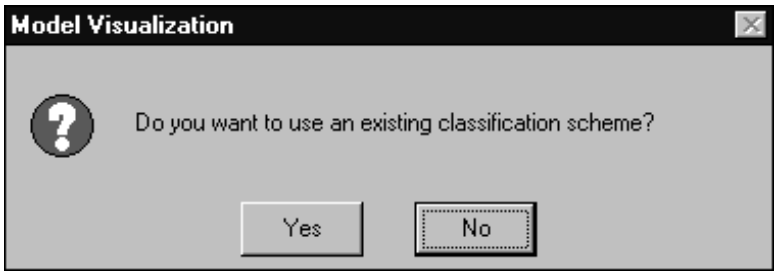
Screen 9.3.2

4. Select the output element you want to visualize in the dialog box, as shown in Screen 9.3.3. Note that the name of the pollutant (e.g., "CBODU (20 deg C), calculated") appears in the text in the dialog box.

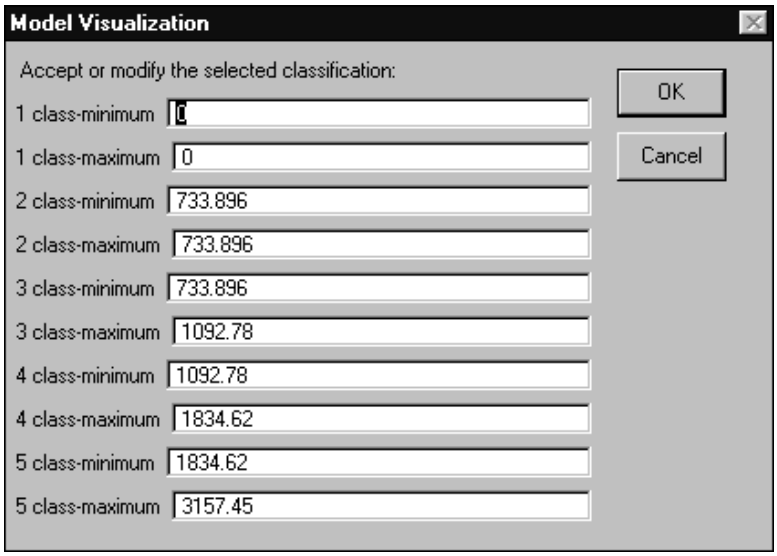


Screen 9.3.3

5. The next dialog box (Screen 9.3.4) asks you whether you want to use previously saved ranges to group results for visual display. If you have not previously selected a scheme or you want to develop a new scheme, click **No**. If you want to use an existing scheme, click **Yes**. You will be prompted to specify the file name of the scheme. Once a file has been selected, a dialog box will prompt you to accept or modify the selected classification (Screen 9.3.5). If needed, make modifications and select OK to continue.

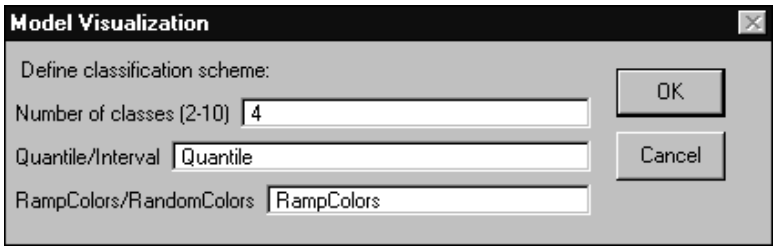


Screen 9.3.4



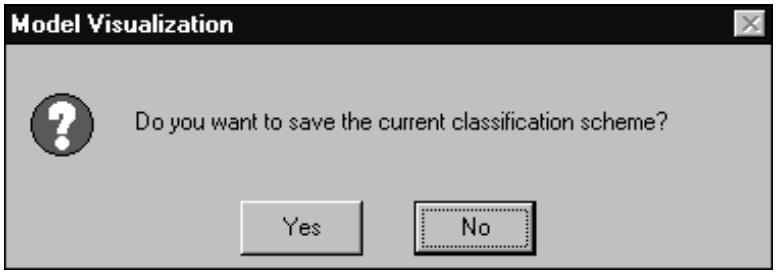
Screen 9.3.5

6. From the classification scheme definition box (Screen 9.3.6), you can select default settings of classes, modify default classes, or edit previous settings of the saved schemes.



Screen 9.3.6

7. The next dialog box (Screen 9.3.7) gives you the option to save the current settings of the classification scheme for future use. If you do not want to save them, click **No**. Otherwise, click **Yes** and you will be prompted to provide a name for the scheme.



Screen 9.3.7

8. Screen 9.3.8 displays the results. One of the two windows has the banner "TOXIROUTE: F=filename, output element name" (e.g., Average Concentration). Notice that the new active theme includes the pollutant name and the legends for the classification scheme. You can double-click with the left mouse button on the window banner for a full screen view.

Visualizing QUAL2E Output

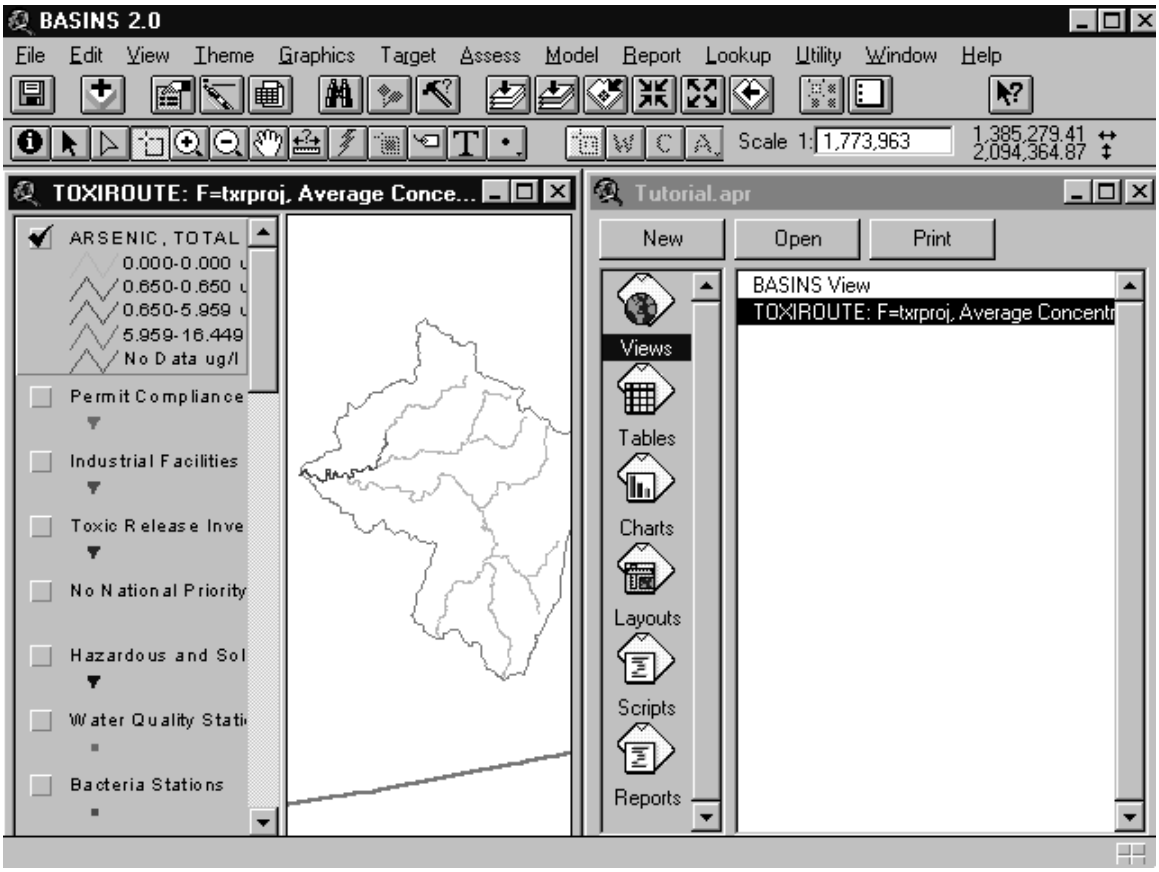
QUAL2E output can include temperature, dissolved oxygen, BOD, algae, nutrients, fecal coliform, and up to three conservative substances and one nonconservative substance. The output file also includes the values of a large number of intermediate variables (or components), which are calculated during the simulation. You might often find it very useful to visualize these variables (e.g., the components of dissolved oxygen balance). Therefore, the pollutant selection dialog box shows a long list of items from which you can choose. ***QUAL2E*** output visualization displays only the reaches that were included in the simulation.

Note: The Windows interface to ***QUAL2E*** can graph model results, but cannot map them.

1. In BASINS View pull down the *Models* menu and select *Visualize*.

2. Choose "QUAL2E output file" in the output file type selection dialog box (Screen 9.3.9).

3. Specify the ***QUAL2E*** output file name in the file selection dialog box (Screen 9.3.10). The ***QUAL2E*** output files are located in the \BASINS\MODELS\QUAL2E directory. There are two ***QUAL2E*** output files. Both have the same name as the ***QUAL2E*** input file (e.g., QAL2E002.INP), but each has a different extension. The summary output file has an .out extension, which is displayed after

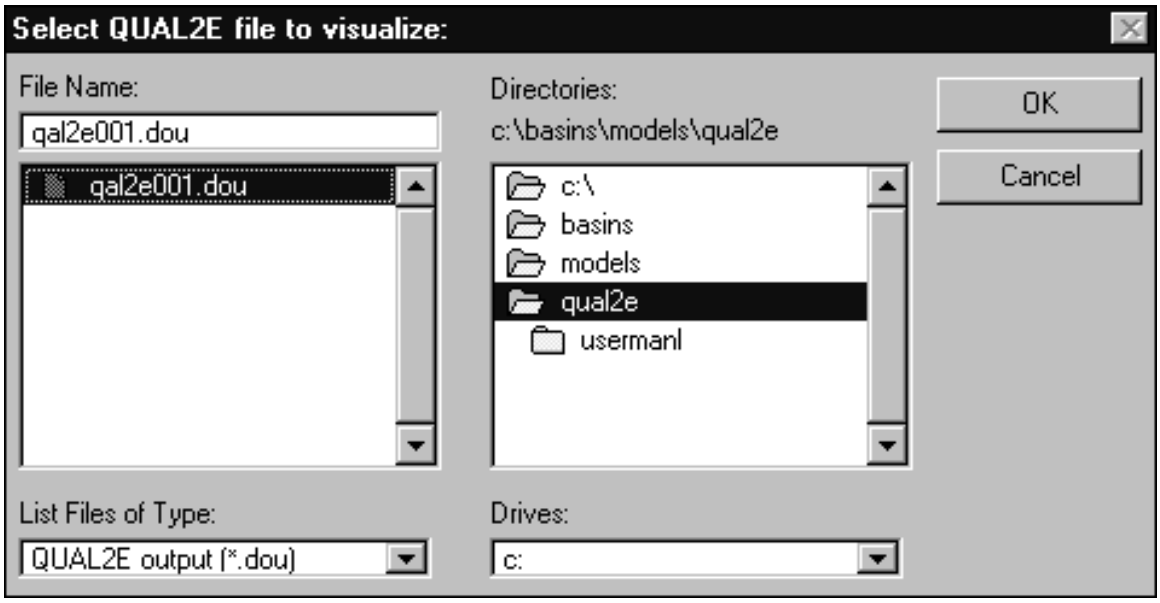


Screen 9.3.8

Screen 9.3.9

each simulation run within the ***QUAL2E*** interface. The other output file has a .dou extension (e.g., QAL2E002.DOU), and it is used in the ***QUAL2E*** output visualization process. Therefore, in the visualize file selection dialog box you will be prompted to choose only the DOU file. Click **OK** after you have selected the output file name.

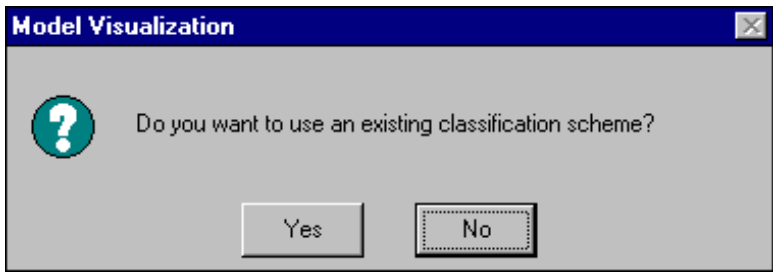
4. Select the output element you want to visualize in the dialog box, as shown in Screen 9.3.11.



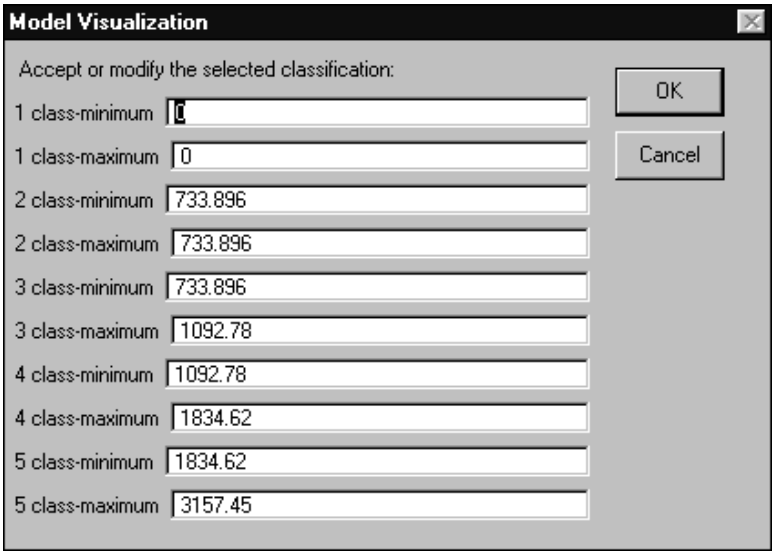
Screen 9.3.10

Screen 9.3.11

5. The next dialog box (Screen 9.3.12) asks whether you want to use previously saved ranges to group results for visual display. Click **No** if you do not have any, or want to create a new presentation scheme. If you want to use an existing scheme, click **Yes** and you will be asked to specify the file name of the scheme. Once a file has been selected, a dialog box will prompt you to accept or modify the selected classification (Screen 9.3.13). If need, make modification and select **OK** to continue.

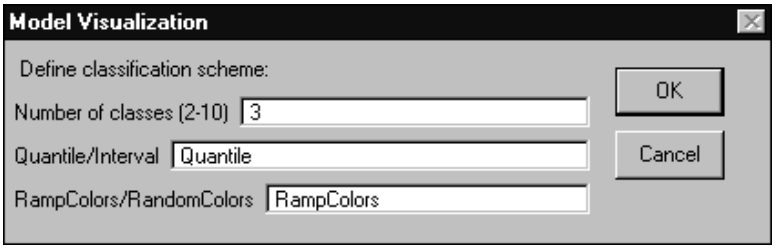


Screen 9.3.12



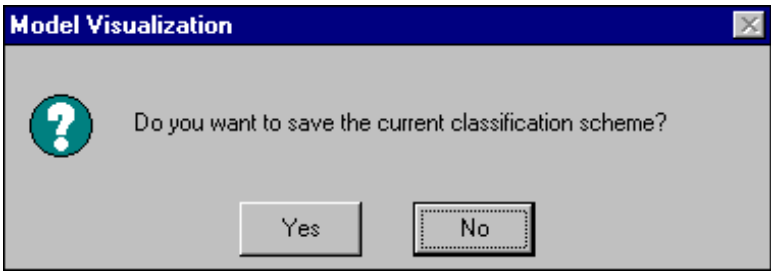
Screen 9.3.13

6. From the classification scheme definition box (Screen 9.3.14), you can select default settings of classes, modify default classes, or edit previous settings of the saved schemes.



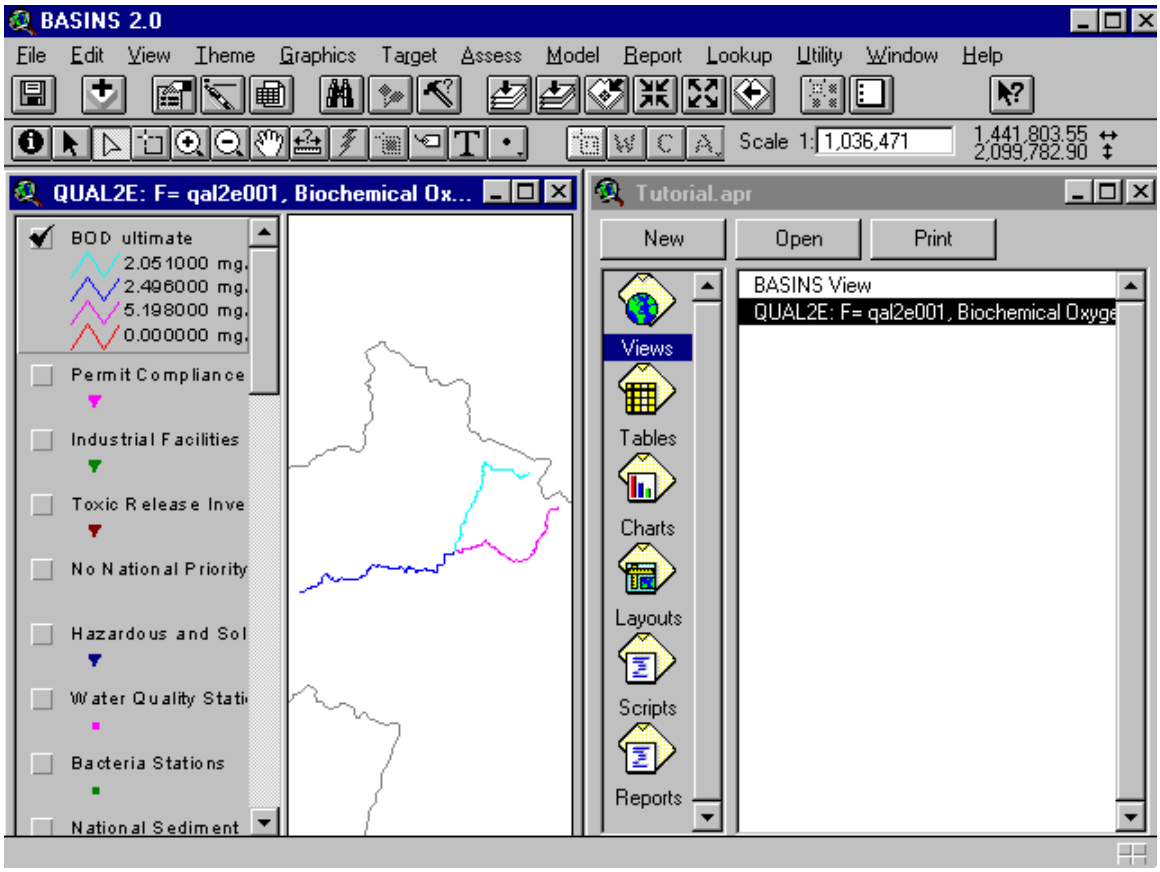
Screen 9.3.14

7. The next dialog box (Screen 9.3.15) gives you the option to save the current settings of the classification scheme for future use. If you do not want to save then, click **No**. Otherwise, click **Yes** and you will be prompted to provide a name for the scheme.



Screen 9.3.15

8. Screen 9.3.16 displays reaches that were included in the QUAL2E simulation each color-coded according to the selected presentation scheme. One of the two windows has the banner "QUAL2E: F=filename, output element name" (e.g., Biochemical Oxygen Demand). Notice that the new active theme includes the selected output element name and the legends for the classification scheme. You can double-click the left mouse button on the window banner for a full screen view. You might want to check the Reach File,V1 theme to view other reaches in the cataloging unit. Other themes can be viewed to examine related information with the model visualization.



Screen 9.3.16